FEM SIMULATION OF FRICTION STIR WELDING PROCESS ON SYSWELD

PROF. DE MADDIS MANUELA
PROF. RUSSO SPENA PASQUALE
PROF. LOMBARDI FRANCO

WANG JINGANG

Academic Year 2020 - 2021
ACKNOWLEDGMENTS

I would like to express my deepest gratitude to my supervisor, Professor Manuela De Maddis. She made instructive comments and suggestions on the research direction of my thesis. During the process of writing the thesis, she gave timely and careful guidance on the difficulties and doubts I encountered, put forward many useful suggestions for improvement. I wish to thank all my colleagues, especially Mr. Panza Luigi, whose assistance was a milestone in the completion of this project.

I wish to show my gratitude to Professor Pasquale Russo Spena and his team to carry out the friction stir welding experiments. Without their tireless help, the project’s goals could not have been achieved.

Thanks to Professor Franco Lombardi for accepting me in the research group and introducing such a professional team to work with me.

Last but not least, thanks my families to support me and encourage me to spend this difficult period.
ABSTRACT

Friction Stir Welding is a solid-state welding technology that uses the thermo-mechanical effect of a rotary tool on the base material to achieve overall welding.

It has been widely used in the welding applications of the low melting point alloy such as aluminium alloy, but for the high melting point and high strength alloys, it is still in the research stage.

With implementation of Friction Stir Welding experiment in laboratory, the aim of this work is to create a virtual simulation of this cutting-edge welding technology. An appropriate process modelling will lead to a good understanding of material flow and temperature field distribution, moreover the prediction of Friction Stir Welding process under different condition will be available. However, it is difficult to numerically simulate this technology because the process is highly complex. Indeed, this is a fully coupled thermomechanical problem with large plastic deformation, heat flow, and non-linearity contact.

In this study, the contact and the friction were predicted using a finite element simulation of the coupled Eulerian Lagrangian formula. Furthermore, within the available range of FEM computing software, the main differences between Abaqus, COMSOL and Sysweld have been discussed. After comparing the calculation methods, complexity of the models, approximation methods adopted and contextualization of the final results, Sysweld software has been chosen to build the numerical model.

The simulation procedure is articulated as model building, model mesh, finite element group definition, process parameter definition, material parameter input, simulation execution and simulation result examination. With the development of simulation procedures, a further research to check the alignment of simulation result and experimental measurements has been carried out.

This numerical model enables the possibility to virtually explore different materials and process conditions, thus speeding up the design phase of advanced joints that require an innovative joining technique such as the Friction Stir Welding process.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td></td>
<td>II</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td></td>
<td>III</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td></td>
<td>IV</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td></td>
<td>VI</td>
</tr>
</tbody>
</table>

## SOLID-STATE WELDING PROCESS ................................. 9

1.1 INTRODUCTION ................................................. 9
1.2 RESISTANCE WELDING ........................................ 9
  1.2.1 Spot welding ......................................... 10
  1.2.2 Seam welding ......................................... 11
  1.2.3 Butt welding ......................................... 11
  1.2.4 Projection welding .................................... 12
1.3 DIFFUSION WELDING ......................................... 13
1.4 EXPLOSION WELDING ......................................... 14
1.5 COLD WELDING ............................................. 15
1.6 FRICTION WELDING ........................................ 16
1.7 ULTRASONIC WELDING ....................................... 17

## FRICTION STIR WELDING PROCESS ............................... 19

2.1 HISTORY .................................................... 19
2.2 THE DEFINITION AND PRINCIPLE OF FSW ....................... 19
2.3 FRICTION STIR WELDING PROCESS ............................... 20
  2.3.1 Joint types .......................................... 21
  2.3.2 Heat input of friction stir welding ................... 21
  2.3.3 Welding parameters .................................. 22
2.4 THE CHARACTERISTICS OF FSW ................................. 22
2.5 APPLICATIONS ............................................... 23

## SIMULATION METHODS OF FSW PROCESS ......................... 25

3.1 INTRODUCTION ............................................... 25
  3.1.1 History of the simulation of welding temperature field | 25
  3.1.2 Different types of computation methods ................. 26
  3.1.3 Heat source simulation ................................ 27
3.2 FINITE ELEMENT METHOD SOFTWARE INTRODUCTION ............ 27
  3.2.1 Process topology ...................................... 28
3.3 DETAILED COMPARISON ....................................... 29
  3.3.1 Modelling process .................................... 29
  3.3.2 Mesh process .......................................... 32
  3.3.3 Parameters ............................................ 32
  3.3.4 Simulation and results ................................ 33
  3.3.5 Summary ............................................... 34

## FSW SIMULATION ON SYSWELD .................................. 35

4.1 MODEL BUILDING AND MESH ................................... 35
  4.1.1 Import the tool ....................................... 35
  4.1.2 Draw main parts ...................................... 37
  4.1.3 Mesh and Collectors .................................. 49
4.2 SIMULATION EXECUTION ........................................................................................................86
  4.2.1 Export the model ................................................................................................................. 86
  4.2.2 Files assemble ...................................................................................................................... 87
  4.2.3 Parameter setup .................................................................................................................. 89
  4.2.4 Material Properties setup .................................................................................................. 92
  4.2.5 Launch Simulation ............................................................................................................. 95
4.3 CHECK RESULTS .................................................................................................................... 97

RESULT ANALYSIS AND FUTURE DEVELOPMENTS .................................................................. 100
5.1 EXPERIMENT PARAMETERS ................................................................................................. 100
5.2 MATERIALS .......................................................................................................................... 100
5.3 SIMULATION RESULTS .......................................................................................................... 100
  5.3.1 Weld speed 50mm/min, Rotation speed 1750 rpm .............................................................. 101
  5.3.2 Weld speed 100mm/min, Rotation speed 1750 rpm .......................................................... 102
  5.3.3 Weld speed 200mm/min, Rotation speed 1750 rpm .......................................................... 103
  5.3.4 Weld speed 100mm/min, Rotation speed 1250 rpm .......................................................... 104
  5.3.5 Weld speed 100mm/min, Rotation speed 2250 rpm .......................................................... 105
5.4 CONCLUSION ......................................................................................................................... 105
5.5 FUTURE DEVELOPMENTS .................................................................................................... 106

BIBLIOGRAPHY ............................................................................................................................ 107
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1.1</td>
<td>CLASSIFICATION</td>
</tr>
<tr>
<td>4.1.2</td>
<td>SPOT WELDING</td>
</tr>
<tr>
<td>3.2.1</td>
<td>OPERATION SEQUENCE OF SIMULATION</td>
</tr>
<tr>
<td>3.2.2</td>
<td>MODEL IN SYSWELD</td>
</tr>
<tr>
<td>3.2.3</td>
<td>MODEL IN ABAQUS</td>
</tr>
<tr>
<td>3.2.4</td>
<td>RESULTS IN SYSWELD</td>
</tr>
<tr>
<td>3.2.5</td>
<td>RESULTS IN ABAQUS</td>
</tr>
<tr>
<td>3.2.6</td>
<td>RESULTS IN COSMOL</td>
</tr>
<tr>
<td>4.1.3</td>
<td>APPLICATION MANAGER UI</td>
</tr>
<tr>
<td>4.1.4</td>
<td>IMPORT TOOL</td>
</tr>
<tr>
<td>4.1.5</td>
<td>TOOL IMAGE</td>
</tr>
<tr>
<td>4.1.6</td>
<td>CURVE</td>
</tr>
<tr>
<td>4.1.7</td>
<td>CIRCLE/ARC</td>
</tr>
<tr>
<td>4.1.8</td>
<td>AXIS DEFINITION</td>
</tr>
<tr>
<td>4.1.9</td>
<td>RADIUS</td>
</tr>
<tr>
<td>4.1.10</td>
<td>BLEND</td>
</tr>
<tr>
<td>4.1.11</td>
<td>PART 3</td>
</tr>
<tr>
<td>4.1.12</td>
<td>PART 3 WITH CAP</td>
</tr>
<tr>
<td>4.1.13</td>
<td>FLUID 2 CURVES</td>
</tr>
<tr>
<td>4.1.14</td>
<td>FLUID</td>
</tr>
<tr>
<td>4.1.15</td>
<td>TWO CURVES</td>
</tr>
<tr>
<td>4.1.16</td>
<td>FLUID</td>
</tr>
<tr>
<td>4.1.17</td>
<td>PART MANAGER</td>
</tr>
<tr>
<td>4.1.18</td>
<td>RENAME</td>
</tr>
<tr>
<td>4.1.19</td>
<td>PART LIST</td>
</tr>
<tr>
<td>4.1.20</td>
<td>FLUID</td>
</tr>
<tr>
<td>4.1.21</td>
<td>BOX</td>
</tr>
<tr>
<td>4.1.22</td>
<td>TRANSFORM</td>
</tr>
<tr>
<td>4.1.23</td>
<td>ENTIRE MODEL</td>
</tr>
<tr>
<td>4.1.24</td>
<td>SPLIT</td>
</tr>
<tr>
<td>4.1.25</td>
<td>SURFACE SPLIT</td>
</tr>
<tr>
<td>4.1.26</td>
<td>SURFACE SPLIT OF SHEET</td>
</tr>
<tr>
<td>4.1.27</td>
<td>SURFACE SPLIT OF THE TOOL</td>
</tr>
<tr>
<td>4.1.28</td>
<td>AUTOMESH SURFACES</td>
</tr>
<tr>
<td>4.1.29</td>
<td>2D MESH</td>
</tr>
<tr>
<td>4.1.30</td>
<td>MESH BOTTOM SURFACE</td>
</tr>
<tr>
<td>4.1.31</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.32</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.33</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.34</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.35</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.36</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.37</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.38</td>
<td>TETRA MESH</td>
</tr>
<tr>
<td>4.1.39</td>
<td>TETRA MESH</td>
</tr>
<tr>
<td>4.1.40</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.41</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.42</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.43</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.44</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.45</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.46</td>
<td>MESH</td>
</tr>
<tr>
<td>4.1.47</td>
<td>MESH</td>
</tr>
</tbody>
</table>
Figure 4.1.97 Delete part ......................................................... 84
Figure 4.2.1 Export the model .................................................. 86
Figure 4.2.2 Save as .ASC file .................................................. 86
Figure 4.2.3 New folder ......................................................... 87
Figure 4.2.4 01 INITIAL_FILES .............................................. 87
Figure 4.2.5 Copy 6 files ....................................................... 88
Figure 4.2.6 Paste files in New folder ...................................... 88
Figure 4.2.7 Parameters in notepad ........................................ 89
Figure 4.2.14 Thermal Conductivity ........................................ 93
Figure 4.3.1 Open Visual-Viewer ........................................... 97
Figure 4.3.2 Open File BUTT_POST1000.ERFH5 ....................... 98
Figure 4.3.3 Check contour results .......................................... 98
Figure 4.3.4 Check Nodes temperature ................................... 99
1 CHAPTER

SOLID-STATE WELDING PROCESS

1.1 Introduction

Solid-state welding, SSW, is a process which the welding temperature is lower than the melting temperature of the base metal and the filler metal, and the pressure is applied for the mutual diffusion of atoms.[1] The solid-state welding method can avoid the occurrence of some phase transformation and reduce the formation of some compounds at the welding interface, so as to enhance the interface bonding strength to the greatest extent.

The solid-state welding can be classified by the types of reaction which will happen during the welding process: Electrical, Chemical or Mechanical.

![Classification Diagram]

**Figure 1.1.1 Classification**

1.2 Resistance welding

Resistance welding is a method of mounting the workpieces to be welded between two electrodes, applying electric current, and heating it to a molten or plastic state by using the resistance thermal effect generated by the current flowing through the contact surface and adjacent area of the workpiece, so as to form metal bonding. [2]

Advantages:

1. When the nugget is formed, it is always surrounded by the plastic ring, and the molten metal is isolated from the air, so the metallurgical process is simple.
2. The heating time is short and the heat is concentrated, so the heat affected zone is small and the deformation and stress are small. Generally, it is not necessary to arrange the correction and heat treatment process after welding.
3. It does not need filler metal such as welding wire and electrode, and welding materials such as oxygen, acetylene and hydrogen, so the welding cost is low.
4. High productivity, no noise and harmful gas, in mass production, can be woven together with other manufacturing processes to the assembly line. However, flash butt welding needs to be isolated due to spark splashing.
5. Simple operation, easy to realize mechanization and automation, improve the working conditions.

Disadvantages:
1. At present, there is still a lack of reliable nondestructive testing methods, and the welding quality can only be checked by the destructive test of process samples and workpieces, as well as by various monitoring technologies.
2. The lap joint of spot and seam welding not only increases the weight of the component, but also forms an angle around the nugget of two plates, resulting in low tensile strength and fatigue strength of the joint.
3. The high power requirement, mechanization and automation of the equipment make the equipment more expensive and difficult to maintain, and the common high-power single-phase AC welding machine is not conducive to the balanced operation of the power grid.

There are four main resistance welding methods, namely spot welding, seam welding, projection welding and butt welding.

1.2.1 Spot welding

Spot welding refers to the welding method of using cylindrical electrode to form a spot between the contact surfaces of two overlapped workpieces. During spot welding process, the workpiece is pressed to make close contact, and then the current is switched on. Under the action of resistance heat, the contact part of the workpiece is melted, and the solder joint is formed after cooling.

Spot welding is mainly used for sheet metal stamping parts with thickness less than 4mm, especially for car body, carriage and aircraft fuselage.
1.2.2 Seam welding

Seam welding refers to the resistance welding method in which the weldment is assembled into lap joint or butt joint and placed between two roller electrodes, the roller pressurizes the weldment and rotates, continuously or intermittently transmits power to form a continuous weld. The circular electrode is used to replace the cylindrical electrode in spot welding and move relative to the workpiece. It is mainly used for welding of oil barrel, can, automobile oil tank, etc. The welding parts are assembled into lap joint or oblique joint and placed between two roller electrodes. The roller pressurizes the welding parts and rotates, and continuously or intermittently transmits power to form a continuous welding line, which is called seam welding. Seam welding is a kind of welding method in which a pair of roller plate electrodes are used to replace the cylindrical electrodes of spot welding and move relative to the workpiece to produce a sealing weld with overlapping nuggets.

Seam welding is widely used in thin plate welding of oil drums, cans, radiators, aircraft and automobile fuel tanks, as well as sealed containers in jet engines, rockets and missiles.

![Figure 1.2.2 Seam welding](image)

1.2.3 Butt welding

Butt welding refers to the method that the weldment is respectively placed between two clamping devices to make its end face aligned, and the electric heating is applied at the contact point for welding. For butt welding, the cross-section size and shape of the contact part of the weldment shall be the same or similar, so as to ensure the uniform heating of the contact surface of the weldment.

Butt welding is mainly used for manufacturing closed parts (such as bicycle rim, steel window, etc.); Rolling material lengthening (such as steel town, steel pipe, steel bar, etc.); Welding of dissimilar materials (such as butt welding of dissimilar materials for saving valuable materials and improving the service life of tool working parts). Butt welding is widely used because of its high productivity and easy automation.
Projection welding is a kind of high efficiency welding method which can carry out multi-point welding at the same time. It can be used to replace arc welding, brazing and biting. The processing speed of this method is fast, and there is no other consumption except power, which is a great characteristic. Compared with spot welding, the difference of convex welding is that the convex points are machined on the plate in advance, or the surface and chamfer on the weldment which can concentrate the current are used as the contact parts during welding. When welding, the convex point contact improves the pressure and current density per unit area, which is conducive to crushing the oxide film on the surface of the plate, concentrating the heat, reducing the shunt, and reducing the center distance of spot welding.

Multi point convex welding can be carried out at one time, which improves the productivity and reduces the warpage of the joint. On the car body, the convex welding nut (the nut with convex points) is generally welded on the thin plate, so that only the bolts need to be tightened during assembly, which improves the assembly efficiency.
1.3 Diffusion welding

Two surfaces are held together under pressure at an elevated temperature and the parts coalesce by solid-state diffusion. It is characterized by high requirements for the surface quality to be welded, long welding time, and unstable joint quality.

Advantages:
1. Due to the overheating and non-melting of the substrate during diffusion welding, almost all metals or non-metals can be welded without reducing the performance of the welded material. It is especially suitable for fusion welding and other materials that are difficult to weld, such as active metals, Heat-resistant alloys, ceramics and composite materials, etc.
For the same materials with poor plasticity or high melting point, as well as dissimilar materials that are not mutually soluble or will produce brittle intermetallic compounds during fusion welding, diffusion welding is a more suitable welding method.

2. The quality of the diffusion welding joint is good, and its microstructure and performance are close to or the same as the base metal. There are no fusion welding defects in the weld, and there is no overheated structure and heat-affected zone. Welding parameters are easy to accurately control, and the joint quality and performance are stable during mass production.

3. The weldment has high precision and small deformation. Due to the small pressure applied during welding, the workpiece is mostly heated as a whole and cooled with the furnace, so the overall plastic deformation of the weldment is very small, and the workpiece after welding is generally no longer machined.

4. Large cross-section workpieces can be welded because the welding pressure is not large, so the tonnage of the equipment required for large cross-section welding is not high and easy to realize.

5. It can weld workpieces with complex structures, inaccessible joints, and large differences in thickness, and can weld many joints in the assembly at the same time.

Disadvantages:

1. The preparation and assembly quality requirement of the surface of the weldment are relatively high, especially for the joint surface.

2. The welding thermal cycle time is long and the productivity is low. Each time welding is as fast as a few minutes and as slow as tens of hours. For some metals, it will cause grain growth.

3. The equipment has a large one-time investment, and the size of the welding workpiece is limited by the equipment, so continuous mass production cannot be carried out.

1.4 Explosion welding
**Figure 1.4.1 Explosion welding**

Explosive welding is a kind of processing technology in which two dissimilar metal plates are welded together by using the explosive energy. There are two forms of spot welding and wire welding, also known as explosive pressure welding. It is mainly used for welding composite plates of reaction tank and storage tank with little change of internal pressure and temperature.

The application fields of explosive welding and explosive composite materials can be summarized into two main aspects:

First, as a new process and technology of metal welding, explosive welding is incomparable to the known welding process and technology. In fact, as long as metal materials have a certain plasticity and impact toughness, they can be explosive welded at room temperature. Even materials with low plasticity and impact toughness can be welded together by thermal explosion.

For the welding of metal materials, the use of new energy will promote the development of welding technology. Explosive welding has become a new welding technology in this field due to the utilization of explosive, a huge energy source.

Second, explosive welding is a new process and technology in the production of metal composite materials, which is its biggest use. Especially after the combination of explosive welding with various pressure processing and machining processes, the production methods and processes of other composite materials will be eclipsed. Explosive welding has obvious advantages in variety, specification, output, quality, market, cost and benefit. Practice has proved that the field of explosive composites is the enrichment and expansion of material science system, explosive composites is a new development direction and one of the frontiers, and it is also an important front army to realize the sustainable development of materials.

1.5 Cold welding

![Figure 1.5.1 Cold welding](image-url)
Cold welding is a kind of method which uses mechanical force, molecular force or electric power to make the welding material diffuse to the surface of the appliance. At the micro scale, this phenomenon is closely related to the diffusion of atoms on the surface of materials.

The cold welding process is carried on the cold welding machine, which is also called ESD. It can be divided into hardfacing cold welding machine, sticking sheet repair cold welding machine and welding copper aluminum wire cold welding machine. The first two are used to repair the wear, scratch, air hole, sand hole and other small defects on the surface of metal and casting; The latter is used for welding copper wire, aluminum wire and other non-ferrous metal wire, such as motor enameled wire.

Due to the difference between cold welding and traditional welding, cold flux has high hardness, adhesion and strength, almost no shrinkage, and can reliably prevent many chemical effects, physical stress and mechanical stress, which is also called "liquid metal". It is one of the main applications of cold welding to connect the same or different materials by using reasonable process and selecting appropriate chemical bonding materials (adhesive, sealant, fixing agent, repair agent, etc.) to realize connection, sealing, fixing and functional coating.

1.6 Friction welding

Friction welding is a kind of welding method which uses the heat generated by the friction of the contact surface of the workpiece as the heat source to make the workpiece produce plastic deformation under the action of pressure.[3]

Under the action of constant or increasing pressure and torque, the friction heat and plastic deformation heat are generated in the friction surface and its adjacent area by using the relative motion between the welding contact end faces, which makes the temperature of the friction surface and its adjacent area rise to a temperature range close to but generally lower than the melting point. The deformation resistance of the material is reduced, the plasticity is improved, and the oxide film on the interface is broken. Under the effect of upsetting pressure, with the plastic deformation and flow of the material, the solid-state welding method is realized through the molecular diffusion and recrystallization of the interface.

Advantages:
1. The welding quality is good and stable. During the welding process, the material to be welded does not melt, and there are no welding defects and joint embrittlement related to melting and solidification. The metal in the welding zone is forged structure. Short welding time and narrow heat affected zone make it easy to obtain good quality welded joint.
2. It can be applied on a wide range of materials. Most of the same and different metals can be welded by friction welding. The dissimilar materials, such as aluminum steel, aluminum copper, titanium copper, intermetallic compound steel, which are difficult to be welded by conventional fusion welding methods, can be welded, and can be used to weld composite materials, functional materials, refractory alloys and other new materials.
3. The welding duration is short with a good productivity. The production efficiency of friction welding is 1 ~ 100 times higher than other welding methods, which is suitable for mass production
4. High precision and low cost for the weldment. The welding deformation caused by welding thermal cycle is small, the dimensional accuracy is high after welding, and there is no need to calibrate and eliminate stress after welding
Disadvantages:
1. The equipment is complex and the cost is huge, so it is only suitable for mass production
2. This welding process is carried out by workpiece rotation which is difficult to weld non-circular cross-section
3. It is difficult to weld the plate-shaped workpiece and thin-walled pipe because these are not easy to clamp
4. This process is limited by the welding machine spindle motor power.

1.7 Ultrasonic welding

![Ultrasonic welding](image)

Figure 1.7.1 Ultrasonic welding

The principle of ultrasonic metal welding is using the mechanical vibration energy of ultrasonic frequency (more than 16KHz). It's a special method to connect the same metal or different metal. When the metal is welded by ultrasonic, it neither transmits current to the workpiece nor applies high temperature heat source to the workpiece, but only transforms the vibration energy of wire frame into the friction work between the workpiece under static pressure. The metallurgical bonding between joints is a kind of solid-state welding without melting of base metal. Therefore, it effectively overcomes the spatter and oxidation phenomenon during resistance welding. Ultrasonic metal welding machine can carry out single point welding of copper, silver, aluminum, nickel and other non-ferrous wire or sheet materials. It can be widely used in the welding of thyristor lead, fuse, electrical lead, lithium battery pole piece and pole lug.

The advantages of ultrasonic metal welding are fast, energy saving, high fusion strength, good conductivity, no spark, close to cold processing; The disadvantages are that the welded metal parts
should not be too thick (generally less than or equal to 5mm), the welding spot should not be too large, and the pressure should be applied.

Ultrasonic metal welding is a kind of mechanical treatment process. In the welding process, there is no current flowing through the weldment, and there is no welding arc such as electric welding mode. Because ultrasonic welding does not have the problems of heat conduction and resistivity, it is undoubtedly an ideal metal welding equipment system for non-ferrous metal materials, it can weld effectively.
2 CHAPTER

FRICION STIR WELDING PROCESS

2.1 History

Friction Stir Welding (FSW) is a non-fusion welding process unlike conventional joining process which are fusion in nature and is derivative of friction welding and produces good quality lap and butt joints. Friction stir welding is a joining process which is developed and patented by TWI Ltd, Cambridge, UK in 1991[4]. TWI filed successfully for patents in Europe, the U.S., Japan, and Australia. TWI then established TWI Group-Sponsored Project 5651,"Development of the New Friction Stir Technique for Welding Aluminum," in 1992 to further study this technique.

The development project was conducted in three phases. Phase I proved FSW to be a realistic and practical welding technique, while at the same time addressing the welding of 6000 series aluminum alloys. Phase II successfully examined the welding of aerospace and ship aluminum alloys, 2000 and 5000 series, respectively. Process parameter tolerances, metallurgical characteristics, and mechanical properties for these materials were established. Phase III developed pertinent data for further industrialization of FSW.

Since its invention, the process has received world-wide attention, and today FSW is used in research and production in many sectors, including aerospace, automotive, railway, shipbuilding, electronic housings, coolers, heat exchangers, and nuclear waste containers.[5]

FSW has been proven to be an effective process for welding aluminum, brass, copper, and other low-melting-temperature materials. The latest phase in FSW research has been aimed at expanding the usefulness of this procedure in high-melting-temperature materials, such as carbon and stainless steels and nickel-based alloys, by developing tools that can withstand the high temperatures and pressures needed to effectively join these materials.

2.2 The definition and principle of FSW

![Figure 2.2.1 Classification of friction welding](image-url)
Friction stir welding method is the same as conventional friction welding, friction heat is also used as welding heat source in friction stir welding. The difference is that the welding process of FSW is completed by a cylindrical welding head (extending into the joint of the workpiece) which rubs against the material of the workpiece through the high-speed rotation of the welding head, so as to increase the temperature of the material at the joint and soften it. At the same time, the material is stirred and rubbed to complete the welding.

FSW joins materials through the use of a non-consumable rotating tool. A typical FSW tool is comprised of two main components: the shoulder and the pin. The rotating tool is plunged at the start of the joint until it is at the appropriate depth where the shoulder contacts the material. The tool then traverses along the weld joint while continuing to rotate. After traveling the desired length, the tool is retracted.

During welding, the material is heated by a combination of frictional and adiabatic heating, plasticizing the base material. This plasticized material is rotated around the tool and consolidated due to compressive forces from the tool.[6] Due to the rotation rate and compressive forces applied by the tool, the base material experiences a high strain rate which promotes dynamic recrystallization. This phenomenon leads to a very fine grain microstructure.

### 2.3 Friction Stir Welding Process

![Friction Stir Welding process](image)

Friction stir welding (FSW) is a kind of solid phase bonding method using friction heat as welding heat source, but it is different from conventional friction welding. During friction stir welding, the weldment is firmly fixed on the working platform. Then, the stir welding head rotates at high speed and the stir welding pin is inserted into the joint of the weldment until the shoulder of the stir welding head is in close contact with the surface of the weldment. The heat generated by the friction between the welding pin and its surrounding base metal and the friction between the shoulder of the welding head and the surface of the weldment will increase the temperature of the material at the joint and soften it.[7] At the same time, the rotating edge of the welding head moves relative to the weldment along the joint, resulting in strong plastic deformation of the material in front of the welding head. As the head moves...
forward, the material with high plastic deformation at the leading edge is squeezed behind the head. A compact solid-phase joint was formed by friction heat generation and forging between the shoulder of the stir head and the surface of the weldment.

2.3.1 Joint types

Friction stir welding can realize the reliable connection between bar and bar, plate and plate. The joints can be designed as butt joint, lap joint, fillet joint and T-joint. It can be used for the welding of circular, circular, nonlinear and three-dimensional welds. Since gravity has no effect on this solid-state welding method, friction stir welding can be used for all position welding, such as horizontal welding, vertical welding, overhead welding, annular rail automatic welding, etc.

Figure 2.3.1 Types of joints

2.3.2 Heat input of friction stir welding

In the process of friction stir welding, the pin rotates at high speed and inserts into the weldment. Then, under the action of welding pressure, the shoulder contacts the surface of the weldment. As a result, a large amount of friction heat is generated between the shoulder and the upper surface of the weldment and between the pin and the joint surface. At the same time, plastic deformation and fluid flow occur near the pin, resulting in deformation and heat generation. Friction heat is the main part of welding heat. As the welding head moves along the weld direction, the heat exerts thermal cycling on the weld and the base metal near the weld, resulting in the dissolution of precipitates in the material and the softening of the weld and heat affected zone. [8] Friction stir welding is essentially a welding method with friction heat as the welding heat source, so the heat input is a direct and key factor affecting the welding quality.

There is an optimum range between the temperature in the welding seam and the mechanical properties of the joint. If it is beyond the optimum range, the heat in and out of the welding seam is too large, and the mechanical properties of the joint will be reduced. Reason: in the welding process of aluminum alloy, the heat effect zone (HAZ) where the
microstructure and properties change on both sides of the welding seam due to the thermal cycle is the main dangerous zone for softening. The width of the softening zone is directly proportional to the heat input. When the temperature in the weld enters the softening temperature of the aluminum alloy, the precipitation and aggregation of strengthening phase will occur in the heat affected zone, the solution strengthening effect of the material will be weakened, and the strength of the weldment will be reduced. With the increase of temperature, the strengthening phase will even occur over aging precipitation, the solution strengthening effect of the material will be worse, and the strength will decrease more.

2.3.3 Welding parameters

Friction stir welding is a complex process. On the premise of the determination of the FSW head, the most important process parameters of FSW are the rotation speed R, welding speed V of the FSW head and the down pressure of the shaft shoulder. The forming characteristics and properties of friction stir welded joints are closely related to the energy absorbed per unit length of weld in the process of friction stir welding. The energy absorbed per unit length of weld is related to the ratio R/V of rotation speed to welding speed. The larger R/V value is, the more times the stir head rotates on the weld with unit length, the more heat input into the material. The higher the temperature of the welding zone is.[9] On the contrary, the lower the R/V value is, the lower the temperature is.

2.4 The characteristics of FSW

Compared with traditional friction welding and other welding methods, friction stir welding has the following advantages[10]

1. The quality of welded joint is high, and it is not easy to produce defects. The weld is extruded in the plastic state and belongs to solid-state welding. Therefore, the joint will not produce some welding defects and embrittlement phenomena related to solidification metallurgy, such as cracks, pores and burning loss of alloy elements. It is suitable for welding colored metals such as aluminum, copper, lead, titanium, zinc, magnesium and their alloys, as well as steel materials and composite materials, It can also be used to connect different materials.

2. Not limited by the shaft parts, it can be used for flat plate butt joint and lap joint, straight weld, fillet weld and girth weld, large frame structure and large cylinder manufacturing, large flat plate butt joint, etc., expanding the scope of application.

3. Easy to realize mechanization and automation, stable quality and high repeatability. Friction stir welding process parameters less, simple welding equipment, easy to achieve automation, so that the welding operation is very simple, the reliability of welding machine operation and welding quality is greatly improved.

4. Low welding cost and high efficiency. There is no need to fill material and protective gas, no need to pre-treat the surface of weldment before welding, and no need to apply protective measures during welding. The edge of thick weldment need not be beveled. It is not necessary to remove the oxide film for welding aluminum workpiece, just remove the oil stain. It is allowed to leave a certain gap during butt joint, and the assembly accuracy is not required.

5. Small welding deformation and high dimensional accuracy of weldment. Because friction stir welding is a solid-state welding, its heating process has the characteristics of high...
energy density and fast heat input, so the welding deformation is small and the residual stress is small. Under the conditions of enough rigidity of welding equipment, accurate assembly and positioning of weldment and strict control of welding parameters, the dimensional accuracy of weldment is high.

6. Green welding. There is no arc radiation, smoke and spatter in the welding process, and the noise is low. Therefore, friction stir welding is a "green welding method" with high quality and low cost.

At the same time, friction stir welding also has some shortages
1. The design, process parameters and mechanical properties of welding tools are only applicable to a small range of alloys with a certain thickness.
2. The wear of the stir welding head is relatively high.
3. At present, the welding speed is not high.
4. Special fixture is needed, and the flexibility of the equipment is poor.

2.5 Applications

1. In the aerospace field, friction stir welding has been successfully applied to the welding of longitudinal butt weld and circumferential lap joint of rocket and space shuttle booster fuel cylinder. The 2014-T6 aluminum alloy with a diameter of 2.4m and a thickness of 22.2mm was welded with a friction stir welding machine called superstir produced by ESAB company. Compared with MIG welding, the defect rate of longitudinal seam of rocket fuel cylinder is very low. There is a defect in the length of MIG welding seam 832cm, while there is a defect in the length of friction stir welding seam 7620cm, which is 1/10 of that of MIG welding. Recently in δ IV rocket. There is no defect in the 1200m long weld of friction stir welding.[11]

2. In the electronic industry, friction stir welding has been used in the welding of large aluminum alloy radiator. Large wide aluminum alloy radiator is an important part of high-speed trains, urban rail and other rail vehicles.

The melting point and thermal conductivity of copper are higher than that of aluminum, so it is very difficult for copper and copper alloy to adopt general fusion welding method. In Europe, a large copper vessel was made by friction stir welding to store high-energy radioactive materials. The cover and the cylinder were welded together. The penetration was 58mm, the shoulder diameter of the stirring head was 60mm, and the temperature near the joint was as high as 750 ℃. Compared with non vacuum electron beam welding, the welding speed of non vacuum electron beam welding is 254 mm/min, while that of friction stir welding is 100.4 mm/min. the welding speed is slower, but the heating input is not increased, so it can meet the requirements.[12]

3. As a mature technology, friction stir welding has been applied to the flat butt joint of Mercedes Benz SL aluminum body and the butt joint of continuous curved surface parts with small curvature radius.

Energy saving and environmental protection is the general trend of automobile manufacturing industry. The most intuitive way to solve the high emission of automobile is to lighten the body. More than 98% of Mercedes Benz SL body parts are made of aluminum. In addition to mechanical connection, TIG fusion welding is also used. Friction stir welding technology is used on the main floor of the body.[13] Among them, weld a and weld B are butt joints of flat plate, the thickness of plate is consistent, the distribution of fish scale is
very uniform, and the back weld mark is very stable, that is, the parameters of connection process are consistent.

With the improvement of people's understanding of friction stir welding technology, it is expected that in the near future, the connection of aluminum alloy materials will be mainly completed by friction stir welding, especially in the launch vehicle, high-speed aluminum alloy train, aluminum alloy high-speed boat, all aluminum alloy automobile and other projects, friction stir welding technology will play a leading role.
3 CHAPTER

SIMULATION METHODS OF FSW PROCESS

3.1 Introduction

3.1.1 History of the simulation of welding temperature field

The essence of welding process is that the material is heated and melted under the action of external heat source, and then solidified under certain cooling conditions. In this process, the change of temperature is the most important factor affecting the welding forming and welding performance. Therefore, it is very important to understand the law of temperature change of material under the action of heat source for studying the welding method and process of material.

As early as the early 20th century, scientists began to study the welding temperature field, and the initial research direction was only limited to the heat conduction law in solid materials.[14] In 1935, D. Rosenthal studied the heat transfer mechanism of moving heat source in solid by using the differential equation of heat conduction.

In 1940s, D. Rosenthal and H. Rykalin obtained the rekalin formula to describe the welding temperature field by simplifying the welding model and using the analytical method. Limited by the scientific research conditions at that time, the material phase transition and a series of other physical and chemical changes involved in the temperature change process were not included in the scope of the model study, The analytical results are often far from the actual values. After that, researchers have done a lot of research and modification on Rosenthal rykalin model[15], which greatly improves the calculation accuracy of welding heat source model. After a lot of experiments, ADAMES, Muyuanbo and Daogeng Daofu have accumulated the measurement data under different parameters and established the welding heat conduction equation under different conditions. Although the workload of this method is large, the results are more accurate than the mathematical analysis method. With the rapid development of computer science, the computing power of computer has far exceeded the limit of human computing power at the beginning of the century. Wilson and Nikcell began to use the finite element method to calculate the parameters of solid heat transfer process.

In 1975, Poley first used computer program to simulate the change of welding temperature field. In 1976, Krutz's paper discussed the possibility of predicting weldability by thermal cycle of temperature field. In the two-dimensional temperature field model, it is assumed that the forward speed of heat source is faster than the heat conduction speed of base metal, and the influence of heat conduction in the forward direction of heat source is ignored in the process of welding simulation. Since then, J. Goldak proposed a new heat source model based on Gauss heat source for arc welding and submerged arc welding. This model considers that the welding energy obeys Gauss distribution, and further studies the flow of molten liquid in the welding pool, so as to make the energy distribution of weld more reasonable in the welding simulation process, and make the welding current, voltage The welding parameters such as welding speed are better connected with the simulation parameters, and the simulation results are closer to the actual welding process. With the continuous improvement of computer performance, the heat source model has gradually developed from two-dimensional to three-dimensional. U. Dilthey et al. established a three-dimensional heat
source model for GMAW of thin plate, and studied the influence of thermodynamics, hydrodynamics and other factors on the welding process. After entering the 1990s, with the development of finite element software, the welding simulation technology gradually matured. Kamel Abderrazak and others used the general finite element software to simulate the welding of magnesium alloy. In 2000, Ronda et al. from the University of Cape Town deduced the phase transformation law and phase transformation plasticity, established the TMM model and formed a unified system. The model not only studies the effect of temperature field on phase transformation, but also studies the effect of stress and strain on phase transformation, which makes the finite element model for the simulation of welding process more mature [16-18].

3.1.2 Different types of computation methods

Due to the need of engineering application, researchers have done a lot of research work. At present, the commonly used solving methods can be divided into three categories: empirical method, analytical method and numerical simulation method. [19,20]

1. Empirical method

In the early stage of empirical method, the prediction of welding temperature field and welding stress field was mainly based on experience and experiment. Through the accumulation of a large number of data, the rule of temperature change and welding deformation were predicted. The accumulation of empirical formula is limited by the use conditions, which can simulate the welding of simple structure to a certain extent. However, in the actual production process, the welding structure is not only composed of simple welding seams. The actual welding seam is often with many passes, many layers and complex joint distribution, so this method is difficult to meet the actual needs.

2. Analytical method.

The analytic method is to form mathematical equation model to describe the actual welding process by the derivation of existing laws and laws. By solving the equation, the analytical solution of the relevant welding model can be obtained. Compared with the empirical method. The physical concept of this method is clear and the logic is rigorous; However, due to the fact that most of the conditions are hypothetical and different from the actual situation, the scope of application is limited, so it is difficult to combine with the actual conditions. For some complex high-order, nonlinear equations, it is often difficult to get the required solution.


The analytical solution of welding model is obtained by calculating the software. This method is called numerical simulation method. At present, the commonly used welding numerical simulation methods include: Monte Carlo method, numerical integration method, finite element difference method and finite element method, in which the finite element method has the advantages of high accuracy and high efficiency, which has been developed into the main research means of welding numerical simulation. For the finite element analysis of welding stress and deformation, the more commonly used methods are inherent strain method and thermoelastic-plastic method.
3.1.3 Heat source simulation

The temperature of welding process changes with time, and the temperature of different areas of base metal is different, which forms the temperature field. Finite element analysis is an important method of welding temperature field simulation. The microstructure and properties, metallurgical process and solid phase transformation in welding process are closely related to heat input and heat conduction. The welding process is a short and fast process, which is usually described as a transient process. The temperature of each area of the welded joint increases rapidly and then decreases rapidly. With the movement of the welding heat source, the temperature field near the heat source changes rapidly with the welding time and relative space position, and some physical and chemical properties of the material itself will also change with the change. This change not only occurs in the corresponding properties, but also produces complex metallurgical phenomena such as metal melting and latent heat of phase transformation at the joint. All these bring difficulties to the numerical simulation of welding process. In order to ensure the correctness of the welding simulation results, the setting of boundary conditions such as the properties of welding materials, the similarity between the welding heat source model and the actual heat source must be consistent with the actual situation.

The establishment of heat source model is an important link in the simulation of welding temperature field. For different welding methods and different heat input, only using the appropriate heat source model can correctly simulate the results consistent with the actual. Although there are many kinds of heat source models, there are three commonly used models: 2D Gaussian model, 3D Gaussian model and double ellipsoid heat source model. In some special cases, due to the actual needs, the combination of the two heat source models will also be used.

The choice of heat source model mainly depends on the welding method and the heat input in the welding process. For surface heat treatment or butt weld with thin plate thickness or material surface heat treatment, 2-gauss model is usually selected. For larger plate thickness direction, 3D Gauss heat source is usually used. However, no matter which kind of Gaussian heat source can not well represent the characteristics of heat source movement, the movement of heat source will lead to uneven heat distribution. If the Gaussian model is still used under the condition of uneven heat distribution, the final result will have a large error with the actual situation, so when the heat source is moving, it is better to use the double ellipsoid model. This is the reason why speed affects the choice of heat source. If the moving speed is very fast, the rising speed of temperature will be much faster than the falling speed, and the heat distribution before and after will be uniform, so the double ellipsoid model must be selected. The heat source model of double ellipsoid is a kind of model with asymmetric front and back distribution. The proportion of heat input is determined according to the size ratio of the front and back ellipses, which can well reflect the phenomenon of uneven heat distribution.

3.2 Finite element method software introduction

The significance of numerical simulation of welding process is to optimize the structure design and process design by calculating the actual situation of repeated welding process, reduce the workload of welding workers, and improve the welding efficiency and welding quality. However, the actual process of welding process is very complex, so the physical and chemical changes of welding process should be fully considered in the numerical simulation.
For example: thermal cycle process in welding process, physical and chemical change process in molten pool transition, behavior of molten pool flow, mechanical behavior of materials, crystallization and phase transformation of materials, growth of grains, segregation of elements, generation and propagation of cracks and many other conditions. In the decades of welding simulation development, scientists have established many welding models. One of the difficulties brought by the gradual improvement of the models is the improvement of the amount of calculation. With the development of computer technology, a large number of commercial finite element simulation software is becoming more and more mature. The commonly used finite element analysis software includes: NASTRAN, ADINA, fluent, ABAQUS, ANSYS, etc. In addition, there are some special software for welding simulation, such as SYSWELD in France, hearts and QUICK WELDER in Japan. In addition, MATLAB and other software can also do a lot of welding numerical simulation work. [21]

The original research of SYSWELD software came from the welding process simulation of nuclear industry. JB. LeBron, a Frenchman, studied the phase transformation of steel during plastic deformation theoretically and numerically. In 1980, French ESI company and famaton company jointly developed SYSWELD software, which further expanded the application scope of SYSWELD software and rapidly promoted its application in aerospace, automobile industry, shipbuilding and other heavy industry fields. SYSWELD software fully realizes the coupling calculation of mechanical process, heat conduction process and metal metallurgy process, and fully considers the metallographic transformation, latent heat of metallographic transformation and metallographic structure change.

3.2.1 Process topology

Here we choose 3 different simulation softwares to compare their process of the simulation:
3.3 Detailed comparision

Regardless of the sequence of the simulation process. All these 3 processes are composed by modelling, mesh, parameters and run simulation&check results. I'll discuss about the differences in these domains.

The process in Sysweld and Abaqus is to simulate the real situation of the friction stir welding process. While in COMSOL, the process is simplified. The model is built as a fixed heat source and moving coordinate. And the welding path is considered as infinite, which ignores the computation around the edges of the sheet.

3.3.1 Modelling process

In Sysweld and Abaqus, the user should import or create the model of tool. While in COMSOL, there is no graphical representation of the tool. The computation only focuses on the pin area and shoulder area affected by the tool.
In COSMOL and Abaqus, the model is made of sheets and tool. While in Sysweld, we need also consider the support elements for the welding process.

In Sysweld and COSMOL, the model include the pin area and melting area (shoulder in COSMOL), which is iconical representation of friction stir welding process. While in Abaqus, you just need to define the face contact between tool shoulder and sheet top face, the pin area overlaps with sheet seems to be fine.

Figure 3.3.1 Model in Sysweld
Figure 3.3.2 Model in Abaqus

Figure 3.3.3 Model in COSMOL
3.3.2 Mesh process

Mesh is the essential procedure in FEM software to run the simulation. The mesh for these 3 methods are more or less the same.
In Sysweld, the mesh is more specific. The user should define the pin and fluid part and name it in collectors. All the boundary conditions, heat transfer and welding parameters are defined by collectors. In this case, the number of collectors are large and all these should be done in mesh procedure which requires the user to have a very good understanding and be very familiar with the process.
In Abaqus, the mesh process is operating contemporarily with the parameter setting process. In this phase, the user is just demanded to mesh the tool and sheet parts. Within parameter setting process, the user will assign the pre-defined material properties to the obtained mesh elements.
In COSMOL, the mesh process is the last before running the simulation. The process is similar with Abaqus, the user only needs to finish the mesh and assign the material to mesh elements.

3.3.3 Parameters

In parameters sets phase, Sysweld will be the most complex and less user-friendly software.
For Sysweld, there is no graphical representation for the user to have a quick understand of the title of input box. User should deal with two document file to filling in the parameters related to welding process, heat transfer and material properties. The only benefit from this method is that as long as the user is fluently grasp the simulation process, he can modify the parameters in a very short time and very efficiently to start a new simulation.
For Abaqus, the parameter setting process is operating contemporarily with the mesh process.
As long as you open the software, it will ask you to fill in the basic numbers to make sure what system of units you are using. And the software has a library to record the material properties you are using. (By user input.) After you create your mesh, you can directly assign the elements with the material in your library. Different from Sysweld, there are no collectors in this software. But more in detail, the software need you to define the boundary conditions, set steps which define the start of the simulation and end of the simulation, and apply load. The user will have a clear view of their inputs and this is easier for beginner to understand.
While in COSMOL, the process is very similar with Abaqus. The user should define the basic parameters, material properties, define steps and boundary conditions.
3.3.4 Simulation and results

The simulation is carried on after the parameters are set.

Figure 3.3.4 Results in Sysweld

Figure 3.3.5 Results in Abaqus
Figure 3.3.6 Results in COSMOL

3.3.5 Summary

The process in Sysweld and Abaqus is to simulate the real situation of the friction stir welding process. While in COMSOL, the process is simplified. The model is built as a fixed heat source and moving coordinate. And the welding path is considered as infinite, which ignores the computation around the edges of the sheet.

In terms of the workload, Sysweld needs the most of work due to a huge number of collectors in the mesh phase. But in Abaqus, with no definition of pin and fluid area in the mesh, probably the computation result will be less accurate.

Considering the re-usability, in Sysweld, the user can quickly modify the parameters and get a new simulation result if the material is changed for example. While in Abaqus, the replacement of the model is easier and less time-consuming compared to Sysweld.
4.1 Model building and mesh

4.1.1 Import the tool

Open Virtual-Environment software. Choose Welding and Assembly, click on Process Executive, Mesh, Assembly, Weld, Heat Treatment, Viewer

Click OK

Figure 4.1.1 Application Manager UI

In Applications screen, click on Mesh.
In **Visual-Mesh** environment, click on **Open file**
Choose your cutting **Tool** obtained from any kind of CAD drawing software with your own specifications. Remember to save your drawing in **.IGS** format
(The rotation axis of your tool should be y axis)

![Figure 4.1.2 Import Tool](image)

**Click Open**

![Figure 4.1.3 Tool image](image)
4.1.2 Draw main parts

Draw PIN

1. Choose Curve, click Circle/Arc

2. In Circle/Arc Window, Click Centre-Axis

Figure 4.1.4 Curve

Figure 4.1.5 Circle/Arc
3. Click **Standard Axis**, along **Y Axis** and **Base Pt 0 0 0**, click OK

![Figure 4.1.6 Axis Definition](image)

4. In Circle/Arc window, input **Radius 4.2**, click **Apply**

![Figure 4.1.7 Radius](image)
5. In Axis Definition window, input **Base Pt 0 2.6 0**, click **OK**
6. In Circle/Arc window, input **Radius 3.7**, click **Apply**
7. Choose **Surface**, click **Blend (Spline)**

![Figure 4.1.8 Blend](image1)

8. Choose these two **new curves**, click your scroll wheel to confirm your choice
9. Input **Part 3**, click **Apply**

![Figure 4.1.9 Part 3](image2)
10. Choose **Surface**, click **Flat (Planar)**

11. Choose the outer curve, input **Part 3** and click **Apply**

![Figure 4.1.10 Part 3 with cap]
**Draw FLUID**
1. Choose Curve, click Circle/Arc
2. In Circle/Arc window, input **Radius 12.5**
3. Click Apply
4. In Axis Definition window, input **Base Pt 0 2.9 0**, click OK
5. Click Apply

![Figure 4.1.11 Fluid 2 curves](image1)

6. Choose Surface, click **Blend (Spline)**
7. Choose these 2 new drawn curves, input **Part 4**, click **Apply**
8. Choose Surface, click **Flat (Planar)**
9. Choose the outer curve, input **Part 4** and click **Apply**

![Figure 4.1.12 Fluid](image2)
10. Choose **Surface**, click **Flat (Planar)**
11. Choose tool cylinder outer curve and Fluid upper curve

![Figure 4.1.13 Two curves](image1)

12. Click **Apply**

![Figure 4.1.14 FLUID](image2)
Rename Part
1. Choose from Part list, right click on 1=>Level_1, choose Part Manager

![Part Manager](image1)

Figure 4.1.15 Part Manager

2. In Part Manager window, Input name TOOL, click Close

![Rename](image2)

Figure 4.1.16 Rename

3. With the same method, rename Part_2 as CURVES
4. Rename Part_3 as PIN
5. Rename Part_3 as FLUID

![Part list](image3)

Figure 4.1.17 Part list
Draw SHEET
1. Hide parts TOOL, CURVES and PIN
2. Choose Volume, click Box

3. In Box window, input Width 3.5, Height 200 and Depth 300, Part 5 and click Apply
4. Choose **Tools**, click **Transform**

![Figure 4.1.20 Transform](image1)

5. In **Transform** window, choose **Surface** and include all **Part 5**
6. Input **dY 0.3**, click **Move**

![Figure 4.1.21 Transform](image2)
Draw SUPPORT
1. Hide FLUID
2. Choose Volume, click Box
3. In Box window, input Width 150, Height 200 and Depth 300, Part 6 and click Apply
4. Hide Part 5
5. Choose Tools, click Transform
6. In Transform window, choose Surface and include all Part 6
7. Input dY 76.75, click Move

Figure 4.1.22 Entire model
Dealing with Details
1. Only show TOOL and Part 5
2. Choose Surface, click Split

3. Include top surface of Part 5, split by Plane

Figure 4.1.23 Split

Figure 4.1.24 Surface Split
4. In **Plane Definition** window, Along **Z Axis**, input **Base Pt 0 0 0**, click **OK**, click **Apply**

![Figure 4.1.25 Surface Split of sheet](image)

5. Show **TOOL**, **CURVES** and **PIN**
6. Choose the **bottom surface** of the tool and split by the pin's outer curve

![Figure 4.1.26 Surface Split of the tool](image)

7. Rename **Part 5** as **SHEET**
8. Rename **Part 6** as **SUPPORT**
4.1.3 Mesh and Collectors

Mesh the Tool
1. Show only the TOOL part
2. Choose 2D, click Automesh Surfaces

![Automesh Surfaces](image1)

Figure 4.1.27 Automesh Surfaces

3. Choose the **bottom surface of the tool**, Set Element Size 0.3,
4. In Method window, choose **Type Tria**. In ID window, input **Part 7**

![2D Mesh](image2)

Figure 4.1.28 2D Mesh
5. Click **Create Mesh**. Click **OK**

6. Select **surrounding surfaces** of the cutter head.
7. In **Advanced** window, click **Within Part** and **Across Part**

---

**Figure 4.1.29 Mesh bottom surface**

**Figure 4.1.30 Mesh bottom surface**
8. In Method window, choose **Pave** and **Graded**
9. Click **Create Mesh**, click **OK**

![Figure 4.1.31 Mesh](image1)

10. Select the adjacent surface of smaller inner circle
11. **Set Element Size 0.4**, in **Method** Window, choose **Auto**
12. Click **OK**

![Figure 4.1.32 Mesh](image2)

13. Select the adjacent surface.
14. **Set Element Size 0.7**, click **OK**

![Figure 4.1.33 Mesh](image3)
15. Select two adjacent surfaces
16. **Set Element Size 1**
17. Reduce the number of edge nodes from 25 to 20, from 35 to 19

![Figure 4.1.34 Mesh](image)

18. In **Edge window**, **Biasing**: choose **Type Linear**, **Bias Edge** to bias the edge with 20 nodes
19. Click **OK**

![Figure 4.1.35 Mesh](image)
20. Select two adjacent surfaces
21. Reduce the number of edge nodes from 25 to 16, from 35 to 13
22. Click OK

![Figure 4.1.36 Mesh](image)

23. Choose the Top surface
24. Click OK

![Figure 4.1.37 Mesh](image)

25. Rename PART_7 as 2D_TOOL
26. Choose 3D, click Tetra Mesh

27. Select all the elements in Part 7
28. In Tetra Mesh window, input Transition Factor 1.3
29. Click Mesh, click Apply

30. Rename Part_8 as 3D_TOOL
Define Collectors 1-3

1. In Selection window, choose 2D Elements, choose Adjacent elements button

![Figure 4.1.40 Adjacent elements](image1)

2. Select the top surface, right click on it Tools, then Add to New Collector

![Figure 4.1.41 New Collector](image2)

3. Right Click on Collector_1, choose Edit

![Figure 4.1.42 Edit Collector](image3)

4. Rename Collector_1 as TOOL_MACHINE
5. In **Selection** window, choose **2D Elements**, choose **Adjacent elements button**
6. Select the **side surface**, right click on it **Tools**, then **Add to New Collector**

![Figure 4.1.43 TOOL_AIR](image1)

7. Rename **Collector_2** as **TOOL_AIR**
8. Select all **3D elements** of part **3D_TOOL**, then **Add to New Collector**
9. Rename **Collector_3** as **M3DO**

![Figure 4.1.44 M3DO](image2)
MESH PIN part & Collector_4
1. Show only part 7 and hide all the top elements

Figure 4.1.45 Part of 2D_TOOL

2. In Views window, choose Right-Side View

Figure 4.1.46 Right-Side View
3. Show also PIN part
4. Hide all the elements not covered by PIN

Figure 4.1.47 Hide the yellow elements

5. Select the remaining elements and Add to New Collector

Figure 4.1.48 Collector_4
6. Select all the elements in **Collector_4**
7. Choose **Tools**, click **Transform**

8. Choose **Copy**, input **Part 9**, click **Copy**
9. Rename Collector_4 as CT_M3DO_M3DP
10. Choose Checks, click Element Normals (Orientation)

![Figure 4.1.51 Check Orientation](image)

11. Choose Arrows, click Flip Normals, choose Elements, select all elements in PART_9

![Figure 4.1.52 Flip Normals](image)

12. Click Apply
13. Show PIN part
14. Choose 2D, click Automesh Surfaces
15. Select side surface of PIN part
16. In 2D Mesh window, Set Element Size 0.3,
17. In Advanced window, click Within Part and Across Part
18. In Method window, choose Type Tria
19. In ID window, input Part 9
20. Click Create Mesh, click OK

![Figure 4.1.53 2D Mesh](image)

21. Select the bottom face of PIN
22. In 2D mesh window, Set Element Size 0.7
23. Click OK

![Figure 4.1.54 2D_PIN](image)
24. Rename **PART_9** as **2D_PIN**

25. Choose **3D**, click **Tetra Mesh**

26. Select all elements in **2D_PIN**, click **Apply**

27. Rename **PART_10** as **3D_PIN**
Define Collectors 5-7
1. Show 2D_PIN part, hide the outer surface, show the same elements as CT_M3DO_M3DP
2. Select all the elements, Add to New Collector
3. Rename Collector_5 as CT_M3DP_M3DO

4. Select all the elements in 3D_PIN, Add to New Collector

5. Rename Collector_6 as M3DP
6. **In Selection** window, choose **Node**

![Selection Node](image1)

Figure 4.1.58 Selection Node

7. **Select all the nodes in 3D_PIN**, Add to New Collector

8. **Rename Collector 7 as M3DP_N**

![3D Model](image2)

Figure 4.1.59 M3DP_N
Mesh FLUID Part & Collector 8-12
1. Show FLUID and CURVES
2. Choose Surface, click Flat (Planar), select 2 circles, input Part 4 and click Apply

3. Show only FLUID and 2D_PIN

5. Select the rest Part of 2D_PIN, Add to New Collector
6. Rename Collector_8 as CTM_M3DP_M3DF

![Figure 4.1.62 CTM_M3DP_M3DF]

7. Choose Tools, click Transform
8. In Transform window, choose Copy, select all 2D Elements in CTM_M3DP_M3DF
9. Input Part 11, click Copy

![Figure 4.1.63 Part 11]
10. Choose **Checks**, click **Element Normals (Orientation)**
11. Choose **Arrows**, click **Flip Normals**, choose **Elements**, select **all elements in PART_11**

![Figure 4.1.64 Flipped Normals of Part 11](image)

12. Select **all elements in PART_11**, Add to New Collector
13. Rename **Collector_9** as **CTM_M3DF_M3DP**
14. Show **FLUID** and **PART_11**
15. Select **adjacent surface**, choose **2D**, click **Automesh Surfaces**
16. In **2D Mesh** window, Set **Element Size** 0.6, Type **Tria**, Method **Auto**, Part **11**
17. Click **Create Mesh**, click **OK**

![Figure 4.1.65 2D Mesh](image)
18. Select adjacent surface, Set Element Size 0.6
19. Click OK

![Figure 4.1.66 2D Mesh](image)

20. Select Side Surface, Set Element Size 0.7
21. Click OK

![Figure 4.1.67 2D Mesh](image)
22. Select bottom surface, Set Element Size 1.2
23. Click OK

24. Choose 3D, click Tetra Mesh
25. In Tetra Mesh window, select all elements in PART_11
26. Click Mesh, click Apply

27. Rename PART_11 as 2D_FLUID
28. Rename PART_12 as 3D_FLUID
29. Show 3D_FLUID
30. Select all 3D Elements, Add to New Collector
31. Rename Collector_10 as M3DF

![Figure 4.1.70 M3DF](image)

32. Show TOOL and 2D_FLUID
33. In Views window, choose Left-Side View. In Selection window, choose Node
34. Select all the visible nodes in 2D_FLUID, except the outer-edge nodes

![Figure 4.1.71 Nodes](image)
35. Add to New Collector your node selection
36. Partly show the bottom surface of 2D_FLUID and Collector_11
37. Select all nodes in bottom surface

38. Right click on 11=>Collector_11
39. Click Add Selection

40. Rename Collector_11 as VZ
41. Show only the side surface of 2D_FLUID
42. In Views window, click Front View. Selection Node
43. Select all nodes in the figure, Add to New Collector

Figure 4.1.74 Collector_12

44. Rename Collector_12 as V_WELDING
Mesh SHEET & Collector 13-14
1. Show CURVES, SHEET and 2D_FLUID
2. Choose Surface, click Split
3. Select sheet top surface, Split by Curves, Select the curve of fluid edge
4. Click Apply

---

5. Delete the selected yellow area
6. Choose Surface, click Split
7. In Surface Split window, Select the whole SHEET part, Split by Plane
8. In Plane Definition window, choose Standard Axis, Along Z Axis, Base Pt 0 0 70
9. Click Apply

---

Figure 4.1.75 Split top surface

Figure 4.1.76 Split top surface
10. Select the whole SHEET part
11. **Base Pt 0 0 25**, click **OK**, click **Apply**
12. **Base Pt 0 0 -25**, click **OK**, click **Apply**
13. **Base Pt 0 0 -70**, click **OK**, click **Apply**

Figure 4.1.77 Split top surface

14. Show outer surface of M3DF, show SHEET
15. Choose **2D**, click **Automesh Surface**, Select the adjacent surface
16. **Set Element Size 1.5**, Type **Tria**, Method **Auto**, Part **13**
17. Click **OK**

Figure 4.1.78 2D Mesh

18. Select the two adjacent surfaces
19. **Set Element Size 2.5**, reduce edge nodes number from **80 to 60**

20. Click **OK**

![Figure 4.1.79 2D Mesh](image1)

21. Select two adjacent surfaces, **Set Element Size 5**, reduce edge nodes number from **40 to 30**

22. Click **OK**

![Figure 4.1.80 2D Mesh](image2)
23. Select 2 middle side surfaces
24. **Set Element Size 1**, reduce edge number **51 to 34**, **Method Map**
25. Click OK

![Figure 4.1.81 2D Mesh](image1)

26. Select 4 adjacent side surface
27. **Set Element Size 1**, reduce edge number **46 to 18**, **Method Map**
28. Click OK
29. Select 4 adjacent side surface
30. **Set Element Size 1**, reduce edge number **81 to 16**, **Method Map**
31. Click OK

![Figure 4.1.82 2D Mesh](image2)
32. Select bottom surface
33. **Set Element Size 1.5, Method Auto**
34. Click **OK**
35. Select the two adjacent surfaces
36. **Set Element Size 2.5**, reduce edge nodes number from **80 to 60**
37. Click **OK**
38. Select two adjacent surfaces, **Set Element Size 5**, reduce edge nodes number from **40 to 30**
39. Click **OK**
40. Select the remaining 2 surfaces, **Set Element Size 1**
41. Click **OK**

![Figure 4.1.83 PART_13](image)

42. Choose **3D**, click **Tetra Mesh**
43. Select all elements in **PART_13** and **outer surface of 2D_FLUID**
44. Click **Mesh**, click **Apply**

![Figure 4.1.84 PART_14](image)
45. Rename PART_13 as 2D_SHEET
46. Rename PART_14 as 3D_SHEET
47. Select all elements in 3D_SHEET, Add to New Collector
48. Rename Collector_13 as M3DT

49. Select top surface and 2 side surfaces which are parallel to X axis
50. Add to New Collectors
51. Rename Collector_14 as M3DT_AIR

Figure 4.1.85 M3DT

Figure 4.1.86 M3DT_AIR
Mesh SUPPORT

1. Choose Tools, click Transform
2. Select bottom surface of SHEET, choose Copy, Part 15
3. Click Copy

4. Choose Checks, click Element Normals (Orientation)
5. Choose Arrows, click Flip Normals, choose Elements, select all elements in PART_15
6. Click Apply
7. Show PART_15 and SUPPORT
8. Choose 2D, click Automesh Surface, choose Front Surface
9. In 2D Mesh window, Set Element Size 15, Part 15, reduce edge nodes from 10 to 6, 20 to 15
10. Click OK

Figure 4.1.89 2D Mesh

11. Select 2 side surfaces, Set Element Size 15, reduce edge nodes from 10 to 6, 13 to 11
12. Click OK
13. Select back surface, Set Element Size 15
14. Click OK
15. Select bottom surface
16. Click OK

Figure 4.1.90 Part_15
17. Choose **3D**, click **Tetra Mesh**
18. Select all elements in **Part_15, Transition Factor 3.5**
19. Click **Apply**

![Figure 4.1.91 Part_16](image)

20. Rename **PART_15** as **2D_SUPPORT**
21. Rename **PART_16** as **3D_SUPPORT**
Collectors 15-
1. Select the bottom surface of 2D_SHEET, Add to New Collector
2. Rename Collector_15 as CT_M3DT_M3DS
3. Select the top surface of 2D_SUPPORT, Add to New Collector
4. Rename Collector_16 as CT_M3DS_M3DT

Figure 4.1.92 CT_M3DS_M3DT

5. Select 2 side surfaces which are parallel to X axis and bottom surface of SUPPORT
6. Add to New Collector, Rename Collector_17 as M3DS_AIR

Figure 4.1.93 M3DS_AIR
7. Select all elements in 3D_SUPPORT, Add to New Collector
8. Rename Collector_18 as M3DS
9. Show 3D_TOOL and 3D_PIN
10. In Selection window, choose Node.
11. Select all nodes in the figure, Add to New Collector
12. Rename Collector_19 as N_ROT

Figure 4.1.94 N_ROT

13. Show 3D_TOOL, 3D_PIN, 3D_FLUID, 3D_SHEET and 3D_SUPPORT
15. Select all the front nodes in X axis positive direction, Add to New Collector

Figure 4.1.95 Collector_20
16. Rename Collector_20 as N_FRONT
17. Show only 2D_PIN and 2D_FLUID without bottom surface or side surface

Figure 4.1.96 Part

18. Show TOOL, choose Left-Side view
19. Delete all the visible 2D elements

Figure 4.1.97 Delete part

20. Selection: Node, select nodes of all the remaining part, Add to New Collector
21. Rename Collector_21 as V_TOOL
22. Select a tip node of the cutter edge, Add to New Collector
23. Rename Collector_21 as NODE_STAB

Figure 4.1.72 NODE_STAB

24. Select all. Choose Tools, click Transform
25. In Transform window, choose Rotate, input Angle 90, choose Move
26. Click Move

Figure 4.1.73 Rotate Part
4.2 Simulation execution

4.2.1 Export the model

From the File menu, choose Export

EXPORT
1. In the Export window, choose Files of type as SYSWORLD ASC data files
2. Give the **File name** as **FSW_DATA1** (as an example)
3. Click **Save**

### 4.2.2 Files assemble

Build a **New Folder** in your working directory

![Folder Menu]

**Figure 4.2.3 New folder**

Find and **Click** the folder **01-INITIAL_FILES** inside the startup directory

![Folder Tree]

**Figure 4.2.4 01-INITIAL_FILES**
Copy the files with name FSW_01, LAUNCH, MAT, parameters, post_tnl.par and thermocouples

Figure 4.2.5 Copy 6 files

Paste the files in your New Folder

Figure 4.2.6 Paste files in New folder
4.2.3 Parameter setup

Open the file parameters

![Parameters file in Notepad](image)

Figure 4.2.7 parameters in notepad

Input values shown in Table 4.2.8
Table 4.2.8 Parameters

<table>
<thead>
<tr>
<th>Name</th>
<th>Value</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>TITL</td>
<td>FSW</td>
<td>The type of model is friction stir welding</td>
</tr>
<tr>
<td>ROOT</td>
<td>FSW_</td>
<td>Root name of the mesh</td>
</tr>
<tr>
<td>MESH</td>
<td>1</td>
<td>Number of the ascii mesh (from example FSW_01)</td>
</tr>
<tr>
<td>AVANce</td>
<td>3.0</td>
<td>Weld speed mm/s</td>
</tr>
<tr>
<td>ROTAtion</td>
<td>68.067</td>
<td>Rotation speed rad/s</td>
</tr>
<tr>
<td>TEMP</td>
<td>20</td>
<td>External temperature</td>
</tr>
<tr>
<td>CONFIG</td>
<td>1</td>
<td>Joint type, 1 Quare butt joint; 2 T joint; 3 Lap joint, see Figure.1.3.2</td>
</tr>
<tr>
<td>INCREment per round</td>
<td>33</td>
<td>Number of triangles based on your mesh, see Figure.1.3.3</td>
</tr>
<tr>
<td>STOP</td>
<td>2</td>
<td>To stop simulation 1 number of rounds; 2 temperature increment</td>
</tr>
<tr>
<td>DELTa Temperature</td>
<td>5</td>
<td>Temperature increment</td>
</tr>
<tr>
<td>MATE</td>
<td>MAT.DAT</td>
<td>File name for material properties storage.</td>
</tr>
</tbody>
</table>

Parameter definition_Joint Type

Figure 4.2.9 Joint Type
The rest of parameters which have not mentioned will remain unchanged.

**Save and Close**

![Image of parameters dialog box](image)

**Figure 4.2.11 Save the data**
4.2.4 Material Properties setup

The material properties can be discovered based on your experimental data, online library or database from Sysweld software. This instruction only introduce how to query from the database of Sysweld software.

Material Database
1. From Database menu, click on Material

![Material Database](Image)

Figure 4.2.12 Material Database

2. In Weld folder, click 5000 Series Al Alloy

![Al Alloy](Image)

Figure 4.2.13 Al Alloy
3. Choose W_AlMgM-Wire-AlMgMn material, click on Thermo-Physical or Mechanical. There are a lot types of properties data available. We can see their specific value under different temperatures.

![Material Database](image)

**Figure 4.2.8 Thermal Conductivity**

**Data Input**

1. **Open** the file MAT.DAT
2. Under the line of Table Conductivity, input 80 / 1 20 0.125 220 0.155 500 0.175 800 0.195 1100 0.205
The numbers mean:
80 means the code number for table conductivity
1 means the data is composed of 1 line
20 0.125 means under 20 °C, the conductivity is 0.125 W/mm°C
220 0.155 means under 220 °C, the conductivity is 0.155 W/mm°C
etc.

We need to modify the values of **conductivity, density, specific heat, strain rate - stress** and **dissipation coefficients**.

By input all the data manually, we assign each properties of our material to the model.

**Save and Close**

---

**Figure 4.2.15 Input the Thermal Conductivity**

**Figure 4.2.16 Save the material properties**
Open the solver SYSWELD (English) Solvers 2019.0 on your pc

Figure 4.2.17 Open SYSWELD Solvers
SYSWELD Solvers

1. In Applications, choose Batch
2. In Products, choose Sysweld
3. In Priority, choose Default
4. Click Multi processors
5. In Working directory, choose Your working directory\New Folder
6. In Input file, choose LAUNCH.DAT
7. Click OK

The calculation of your model will start immediately.
4.3 Check results

Open the software Visual-Viewer

Figure 4.3.1 Open Visual-Viewer
Visual-Viewer
1. Click **Open File**, choose Your working directory\New Folder\Butt_POST1000.erfh5

![Figure 4.3.2 Open File Butt_POST1000.erfh5](image)

2. Click **Results**
3. In the **Results** Window, click **Contour**

![Figure 4.3.3 Check contour results](image)
4. In the Contour Window, click Temperature_NOD or VELOCITY_NOD to see detailed temperature and velocity field.

Figure 4.3.4 Check Nodes temperature
5 CHAPTER
RESULT ANALYSIS AND FUTURE DEVELOPMENTS

5.1 Experiment parameters

<table>
<thead>
<tr>
<th></th>
<th>Unit</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weld speed</td>
<td>[mm/min]</td>
<td>50</td>
<td>100</td>
<td>200</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>[rpm]</td>
<td>1750</td>
<td>1750</td>
<td>1750</td>
<td>1250</td>
<td>2250</td>
</tr>
<tr>
<td>Ambient Temperature</td>
<td>[°C]</td>
<td>28</td>
<td>28</td>
<td>28</td>
<td>28</td>
<td>28</td>
</tr>
<tr>
<td>Vertical force</td>
<td>[kN]</td>
<td>1.7</td>
<td>1.8</td>
<td>2.15</td>
<td>2.1</td>
<td>1.5</td>
</tr>
<tr>
<td>Tmax at the sample surface close to the tool</td>
<td>[°C]</td>
<td>440</td>
<td>367</td>
<td>332</td>
<td>310</td>
<td>368</td>
</tr>
</tbody>
</table>

Table 5.1.1 Experiment parameters

As what we have mentioned on 4.2.4, we have carried out several different simulations to verify what we have obtained from our lab experiment. We simply modify the parameters which have been listed in Table 5.1.1, we can obtain our simulation results.

5.2 Materials

For our welding tool which is rotating above the weldment, we use the Tungsten/Rhenium alloy (75% W, 25% Re).

For our lap joint, we use a Aluminum alloy 6016 grade - DC05 steel (very low carbon steel). Aluminum sheet at the tool side, steel at the worktable side.

For the support which is used as a worktable, we use material 39NiCrMo3.

5.3 Simulation results

In the figures of simulation results. The top half sheet is Aluminum alloy 6106 while the bottom half sheet is DC05 steel. Aluminium 6106 has a higher thermal conductivity and higher specific heat compared to DC05 steel.

Under the simulation conditions, the sheet is considered as infinite length along the welding direction. And this simulation process will stop only if the temperature variation is smaller than 5 degrees. So in this case, we have the same starting phase for all these 5 groups of experiment. Starting from the room temperature 28 degrees, we will only check the temperature field distribution in the end phase.
5.3.1  Weld speed 50mm/min, Rotation speed 1750 rpm

From the figure 5.3.1, we can see that the maximum temperature is in the node 11725 at tool edge. And the maximum value of temperature is 397.4 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side. The simulation takes 165 steps to reach this end phase.
5.3.2  Weld speed 100mm/min, Rotation speed 1750 rpm

From the figure 5.3.2, we can see that the maximum temperature is in the node 11697 at tool edge. And the maximum value of temperature is 396.2 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 1, the maximum temperature reduced a little bit. But it takes much less steps to reach this end phase. In group 2, it takes only 116 steps.
5.3.3 Weld speed 200mm/min, Rotation speed 1750 rpm

Figure 5.3.3 Temperature field of group 3

From the figure 5.3.3, we can see that the maximum temperature is in the node at tool edge. And the maximum value of temperature is 398.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 1 and 2. The maximum temperature is almost the same. But we can see from the contours of the temperature field. Because of the short time to reach its end phase. The temperature is conducted much slower in group 3 than in group 1 and 2. Take the node at middle left point of the sheet, in group 1 it's in the temperature range of 70 degrees. While in group 2 and 3, it's in the temperature range of 45 degrees.
5.3.4  Weld speed 100mm/min, Rotation speed 1250 rpm

From the figure 5.3.4, we can see that the maximum temperature is in the node at tool edge. And the maximum value of temperature is 379.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 2, with the same weld speed but group 4 has a slower rotation speed. In this case, group 4 takes a longer time to reach the end phase, which has 196 steps. At the end phase, the maximum temperature is a little bit lower.

But the temperature field distribution is the more or less the same as group 2.
5.3.5 Weld speed 100mm/min, Rotation speed 2250 rpm

From the figure 5.3.5, we can see that the maximum temperature is in the node 11710 at tool edge. And the maximum value of temperature is 450.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 2 and group 4, group 5 has a much higher rotation speed. And it leads to a much higher maximum temperature. In this case, it takes longer time to reach the end phase.

### 5.4 Conclusion

<table>
<thead>
<tr>
<th></th>
<th>Unit</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tmax in experiment</td>
<td>[°C]</td>
<td>440</td>
<td>367</td>
<td>332</td>
<td>310</td>
<td>368</td>
</tr>
<tr>
<td>Tmax in simulation</td>
<td>[°C]</td>
<td>397.4</td>
<td>396.2</td>
<td>398.9</td>
<td>379.9</td>
<td>450.9</td>
</tr>
</tbody>
</table>

Table 5.4.1 Maximum temperature

Comparing the results obtained from the simulation and in the experiment, we can see a difference in the values.

The reason is probably that: in the experiment, the welding sheet has a fixed length along the welding direction, while in the simulation the length is considered as infinite. In this case, in the experiment, the maximum value of the temperature may have not been reached before
the welding tool has finished the weldment. And in the simulation result, the process can be relatively long enough for the temperature to reach its maximum.

When the welding speed is increasing, the maximum temperature is decreasing in experiment. Because the duration of the welding process is reduced, so the maximum temperature may have not been reached.

When the rotation speed is increasing, the simulation results show that, the maximum temperature will increase significantly. Because the faster rotation speed increase the heat generation in the contacting face, which leads to a higher temperatures.

5.5 Future developments

By using SYSWELD, I have obtained an amazing result to simulate the friction stir welding process. But during my experience of using this software, I can feel that there need to be more developments for this software to improve the user experience.

From the simulation results, we can view the node temperature and velocity field. But for the further thermo-mechanical results, we would like to have those datas either. During the model building phase, I was always worried about having made any mistakes, and I need to save a copy of current state every few minutes. Because the software does not support the undo command. For the time of setting parameters, it's always hard for me to define the strain rate of a specific material, which is very unrare to look up online and I would like to have some default values of common material for the users to choose.
BIBLIOGRAPHY


